Intusoft Newsletter

Personal Computer Circuit & System Design Tools



Issue #46 May 1996
Tel. (310) 833-0710
Fax (310) 833-9658

New Magnetics Designer

Intusoft has released a new software program for transformer and inductor design. The program is called Magnetics Designer. **Magnetics Designer synthesizes and analyzes various types of transformers, inductors, and chokes.** It is optimized to design layer wound magnetics using cores that have rectangular or circular winding areas. Typical designs may include ferrite pot cores, EI, EE, EP, cut C, and RM cores, toroids, and EI laminated cores. Magnetics Designer is capable of designing single phase multi-winding transformers and inductors operating from 10Hz to well above 1MHz. Typical applications include 60 Hz line transformers, switching regulator transformers, and output filter inductors.

Continued on pg. 2

In This Issue

2 **New Magnetics Designer Released**
5 Generating SPICE Models For Magnetics
8 New IsSpice4 Has New BSIM3v3 MOS Model
10 Pspice Abandons Users Again
11 Modeling Mechatronics: Synchro Modeling
14 Simply The Best Tools

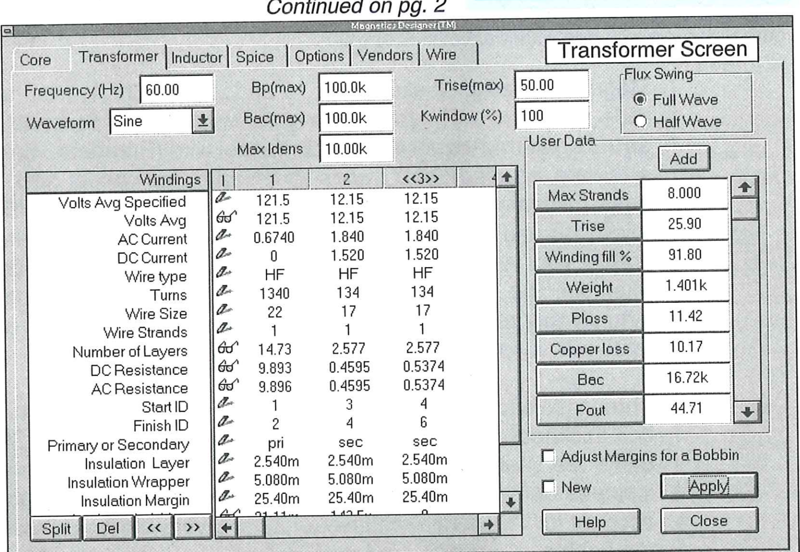


Figure 1, The new Magnetics Designer program makes transformer and inductor design easy. A demo and FAQ are available at http://www.intusoft.com.

New Magnetics Design Software

A database with thousands of cores and a wide variety of materials is included with the program. They are accessed through the Core screen shown in Figure 2. Vendors like TDK, Magnetics, Philips, Thomson, Micrometals, and Ferrite International are represented. However, the database is extensible. **You can add your own core and material information using a supplied Excel spreadsheet template.**

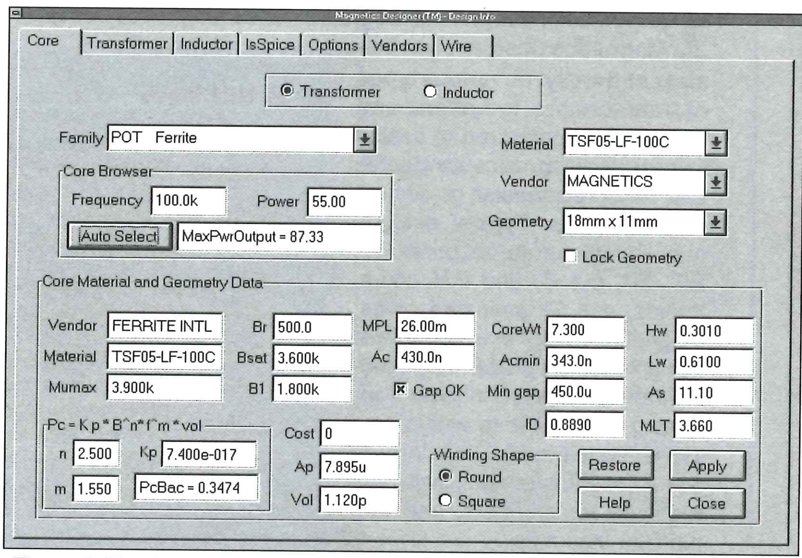


Figure 2, The Core Screen allows you to browse through and change the extensive core database included with Magnetics Designer.

Magnetics Designer allows you to choose (Figure 1) from a wide variety of wire types: **heavy formvar, small formvar, square, double-square** and **Litz** magnet wire (European wire sizes are included); **pcb traces** specified by the copper thickness, minimum trace width, minimum trace spacing, and PWB thickness and **foil** specified in various thicknesses. You can create your own wire tables with different wire sizes. For example, the square wire table can be edited to describe virtually any rectangular wire format, or you could describe a table with a sparse set of gauges (i.e., even gauges only).

Three insulation options allow the user to specify layer insulation thickness, wrapper thickness, and end margin lengths. If there are multilayer foil windings, layer insulation may be included between each layer of foil. There is also an option to account for the margin required for inner layer windings to exit the bobbin.

How Magnetics Designer Works

To start a transformer design, you first select the core family and material on the core screen (Figure 2). Then the operating frequency and electrical specifications (voltage and current per winding) are entered on the transformer screen (Figure 1). You may optionally enter one or more design constraints like temperature rise, peak flux density, and/or window fill.

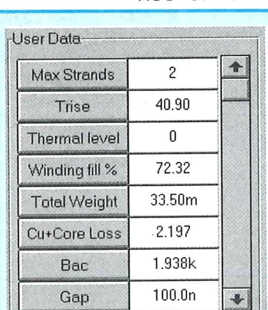
Magnetics Designer will optimize the design using a powerful set of state-of-the-art algorithms. The program will select the core geometry, turns ratio, wire sizes, and number of strands to achieve your design goals, all while minimizing the total power dissipation for a given temperature rise in the smallest possible core.

Once finished, Magnetics Designer will provide a summary of the cores it tried to optimize, and suggestions on how to possibly improve the design if the design constraints were not met. You may browse through the core database and choose an alternate core geometry. If desired, you can lock your choice and optimize the design around your selection.

As shown in the transformer screen, you can manipulate over 20 parameters such as the wire size and type, number of strands, gap length, insulation thickness and more. Magnetics Designer will predict characteristics like peak flux density, AC flux density, DC and high frequency AC resistance, core and copper losses, leakage inductance, winding capacitance, weight, efficiency, temperature rise, and window fill percentage.

Program Sports Unique Options

Magnetics Designer has several unique features. The program exposes the mathematical equations and algorithms it uses to calculate the transformer or inductor performance (Figure 3). For example, you can access a parameter such as temperature rise or leakage inductance and view the equation used to



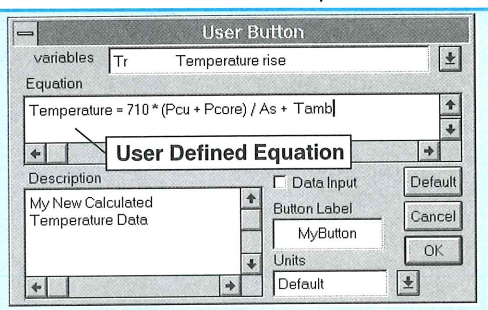


Figure 3, Magnetics Designer outputs a variety of data including temperature rise, weight, and fill in the "User Data" Fields (left & Figure 1). The User Button dialog allows you to edit the equation(s) which defines each field.

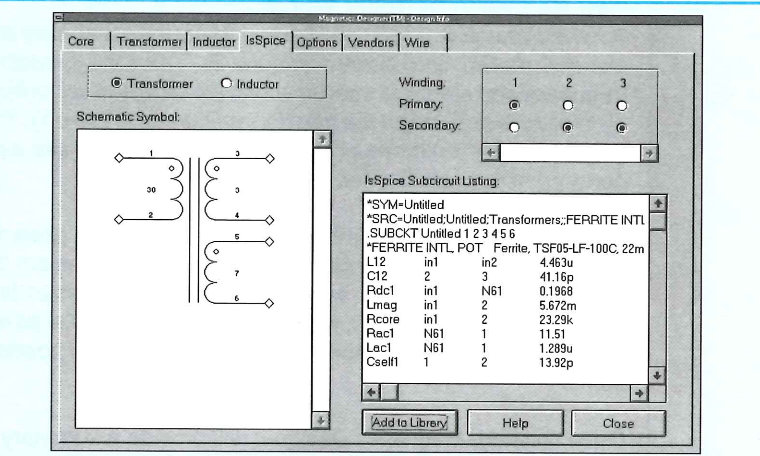


Figure 4, Magnetics Designer makes SPICE models for magnetics. You can configure and edit a schematic symbol in the IsSpice screen shown above.

calculate its value. You can then substitute one or more expressions in place of the default equation. Equations can be set up using data from the program to derive your own performance merits. <u>This feature allows Magnetics Designer to be customized for each application or design.</u>

Another unique function is the SPICE modeling capability. Magnetics Designer produces a Berkeley SPICE model of your transformer or inductor. The program also allows you to produce and configure (Figure 4) schematic symbols. The symbols can be used with schematic entry packages from OrCAD (CAPTURE®), Protel (Schematic³) or Intusoft (SpiceNet).

Within the general constraints described above, you can see that Magnetics Designer has a great deal of flexibility and can

1:1 Xfrm, Magnetics EC35 Core using Ferrite - F material characterized at >100kHz and 100 degrees C

Weight (Grams):	36.00	
Window Fill (%):	30.59	
Magnetizing Ind. (Henry):	1.024m	
Pk. Flux Density (Gauss):	3.498k	
Core Loss (Watts):	1.389	
Temp. Rise (deg C):	23.32	
Winding Number:	1	2
Volts Average (V):	100.0	100.0
AC Current (A):	0.1000	0.1000
AC Resistance (Ω):	0.1846	0.2052
DC Resistance (Ω):	67.26m	74.78m
Copper Loss (W):	1.846m	2.052m
Wire Type/Awg:	HF/22	HF/22

```
.SUBCKT ec35 1 2 3 4
L12      in1   in2      407.2n
C2_12      2     4      189.5p
C3_12      2   N42      -41.20p
C1_1     in2     2      72.33p
Rdc1     in1   N61      0.1120
Lmag     in1     2      924.0u
Rcore    in1     2      4.943k
Rac1     N61     1      1.137
Lac1     N61     1      121.5n
Efwd2    N824  in2   2  1.000
Vsens2   N82   N42
Ffbk2    in2   2  Vsens2  1.000
C1_2     N82     4      3.085p
Rdc2     N42   N62      0.1200
Rac2     N62     3      1.219
Lac2     N62     3      130.2n
.ENDS
```

Figure 5, The SPICE netlist models the core, AC/DC resistance, core loss, leakage inductance, and winding capacitance. Some design characteristics are also shown.

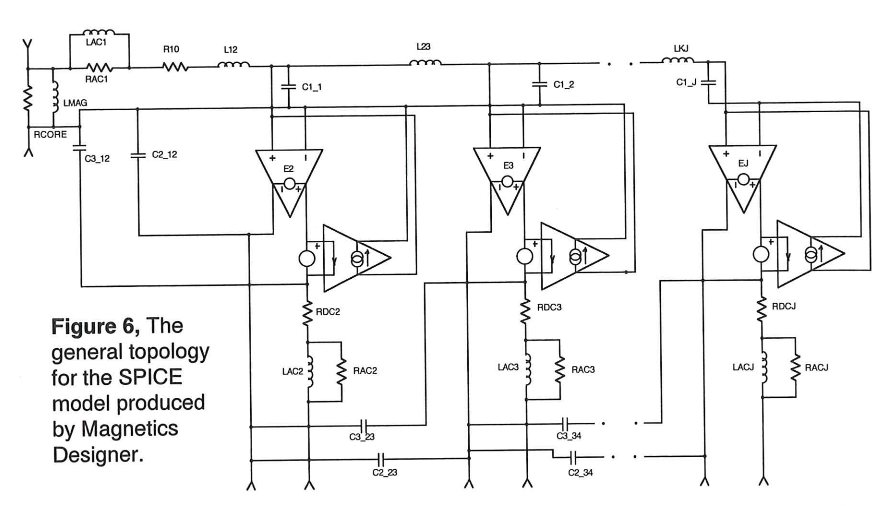


Figure 6, The general topology for the SPICE model produced by Magnetics Designer.

handle many different winding configurations, wire types, and core configurations.

How Magnetic Models Are Generated

The complete SPICE model for a layered winding is shown in Figure 6. Notice that the AC resistance is approximated using a single L-C parallel section. The impedance is fit to the AC resistance curve in the neighborhood of resonance. A plot of AC resistance is shown in Figure 7. For all cases we have observed, the leakage inductance produces a much higher impedance than the AC resistance. This leads us to conclude that the effect of the AC resistance will be to damp the resonance of the transformer, hence the importance of making the model accurate in the neighborhood of the expected resonance. The resonant frequency using leakage inductance (L12, etc.) and winding capacitance represents a theoretical maximum. We reduce this frequency slightly by summing the previous leakage

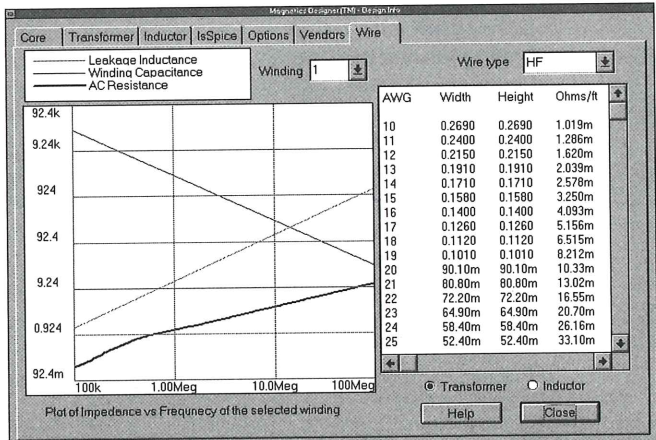


Figure 7, The wire screen provides insight into the expected losses due to harmonics with a plot of impedance vs. frequency for each winding.

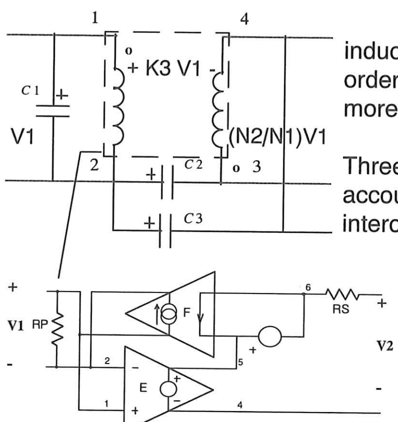


Figure 8, Capacitance and Ideal transformer topologies.

inductance and inter-winding capacitance in order to align our AC resistance calculation more closely with expected circuit operation.

Three capacitors (Figure 8) are sufficient to account for the switching energy for different interconnection schemes. Certain turns ratios and winding directions result in negative capacitor values. IsSpice handles this properly, as long as the negative capacitance does not "see" a real valued resistor. Therefore, the model topology is restricted to placing the inter-winding capacitances "inside" of the winding resistance and leakage inductance models. As with most power electronics simulations, Gear integration works best. IsSpice users can switch to Gear integration by issuing the .Options parameter Method=Gear. Many other SPICE programs, such as Pspice and Electronic Workbench, do not let you force Gear integration, therefore, greatly limiting their effectiveness for power electronics simulation.

The ideal transformer accounts for the forward voltage transfer and reverse current transfer, and is modeled using a pair of IsSpice controlled sources (E & F). In the forward direction, the "primary" voltage is transferred to the secondary using a voltage controlled voltage source with its gain set to the turns ratio. In the reverse direction, a voltage controlled current source assures that current is reflected from the secondary to the primary by the turns ratio. The voltage source after the first winding is used to sense current for the feedback term in the ideal transformer model. The DC and AC resistance equivalent circuits are each attached to their respective windings.

The core is modeled as a simple resistor in parallel with the magnetizing inductance. The resistance is set by the core loss equations at the operating frequency. A linear core model is used. In the June 1995 *Intusoft Newsletter*, we illustrated techniques for making a nonlinear saturable core model which can be added. The eddy current break is not sufficiently described in the program's core database, so its effect isn't modeled. This effect can be modeled by adding a series R-C network across the magnetizing inductance, set to match manufacturer's data. This could add extra damping to the ringing observed when switching devices turn off while still conducting current. Modeling added effects requires considerable computational resources, and shouldn't be done unless you depend on the nonlinearities for an aspect of your design.





The Magnetics Designer software was originally developed by **Analytic Artistry**, a leader in the field of nonlinear magnetics design software. In 1995, Intusoft acquired the rights to the "Transformer and Inductor Spreadsheets", two separate DOS based programs. A large number of changes have been made, including combining of the DOS programs into one unified solution. Intusoft is now the sole owner and developer of the Magnetics Designer software. A brief list of enhancements over the DOS version is shown next:

- The core selection and transformer/inductor design algorithms have been changed. Magnetics Designer now automatically tries the suggestions previously shown in the documentation. You can switch cores, families, geometries and materials at any time and lock your choice to play "what-if" investigations.
- IsSpice models and schematic symbols are automatically generated.
- Planar Magnetics (Printed circuit board cores and wire traces) are included.
- You can define your own core materials, geometries, and wire tables. Wire tables may have sparse gauges, i.e., you can have an even gauge table.
- Leakage inductance and winding capacitance values are calculated.
- The split winding feature is automatic. You can clone and move windings around and the proximity fields are recalculated based on the interleaved configuration.
- AC resistance is computed for square wave harmonics (up to the 100th harmonic) or for a pure sine wave.
- You can set the minimum number of turns for the winding with the smallest turn count. You can control the accuracy of the secondary voltage outputs.
- Three thermal models are supported. The thermal model allows user entry of the convection coefficient as well as calculation of the hotspot temperature in the winding stack, or core, based upon heat flow through the windings or insulation to the core. Other models may also be implemented.
- The program supports alternate sets of units and European wire sizes.
- Magnetics Designer is compatible with Windows 3.1x, 95, or NT.

Documentation, Pricing, and Availability

A detailed applications manual is included with the program. It provides information for the engineer who wants to learn more about magnetics design. It describes the design equations and algorithms, discusses core materials and geometries, and presents detailed examples. Virtually all of the program's documentation is also available on-line.

As a Special Introductory Offer, Magnetics Designer is discounted until Aug. 30, 1996. (excluding North America) After that date, the price will be increased. Contact your local Intusoft Dealer for more information. More information, including an application note and a demonstration version of Magnetics Designer, is available on the Intusoft Web Site http://www.intusoft.com and CompuServe (CADD/CAM/CAE Vendor forum, Library 21).

IsSpice4 Gets New BSIM3 Model

By Farhad Aryom

MOS IC technology has made tremendous progress in recent years. The desire to increase circuit speed and reduce circuit area has continually reduced the feature size of MOS ICs. As size decreases, short channel effects and high field effects, which did not exist in large devices, become more and more significant, causing device characteristics to deviate substantially from long channel device behavior. In 1983, the Berkeley Short-Channel IGFET Model (BSIM) version 1 was developed. BSIM1 works reasonably well for devices with channel lengths of 1.0μm or larger. In 1989, an enhanced version, BSIM2, was developed. BSIM2 can fit devices down to 0.25μm for both current, as well as small signal parameters such as output resistance.

The parameter extraction strategy used for BSIM1 and BSIM2 is relatively simple and accurate. But it introduces some inherent problems. First, a single process file does not work well when a wide range of device dimensions are used. Multiple process files are needed to obtain good accuracy over the full range of device dimensions. Second, the existence of redundant parameters makes statistical circuit performance prediction difficult, if not impossible. BSIM3v3 is the latest deep-submicron MOSFET model from the University of California at Berkeley, and it is designed to address these problems.

BSIM3 is a physical model and is based on a coherent quasi two-dimensional analysis of the MOSFET device structure, taking into account the effects of device geometry and process parameters. In other words, dependencies of important geometry and process parameters such as channel length, channel width, gate oxide thickness, junction depth, substrate doping concentration, etc. are built into the model. BSIM3 allows users to accurately model MOSFET behavior over a wide range of existing technologies and predict the behavior for future technologies.

The BSIM3v3 (version 3) model has been extensively modified from previous releases and is the recommended version. Some of the modifications that are not found in version 2 are:

- A single I-V expression is used to model current and output conductance characteristics for subthreshold, strong inversion, linear, and saturation regions. This formulation guarantees continuity for **Ids, Gds, Gm** and their derivatives for all **Vgs** and **Vds** bias conditions.

- There are new width dependencies for bulk charge and source/drain resistance, **Rds**. This greatly enhances the accuracy of the model for narrow width devices.

- **dw** and **dl** dependencies are available for different Wdrawn and Ldrawn devices. This improves the model's ability to fit a variety of **W/L** ratios with a single set of parameters.

- New capacitance equations improve the modeling of short and narrow geometry devices.

- New relaxation time model for characterizing the non-quasi-static effect of MOS circuits for improved transient behavior.

Simulation Characteristics

The various DC characteristics of a typical short channel BSIM3 model are shown in Figure 9. The op-amp circuit (Figure 10) uses a BSIM3 model consisting of all the default parameter values. The frequency response for this circuit is shown below. A transient simulation of a one-bit comparator, along with the previous simulations, is also included on the *Intusoft Newsletter* subscriber's disk, in addition to several other test circuits.

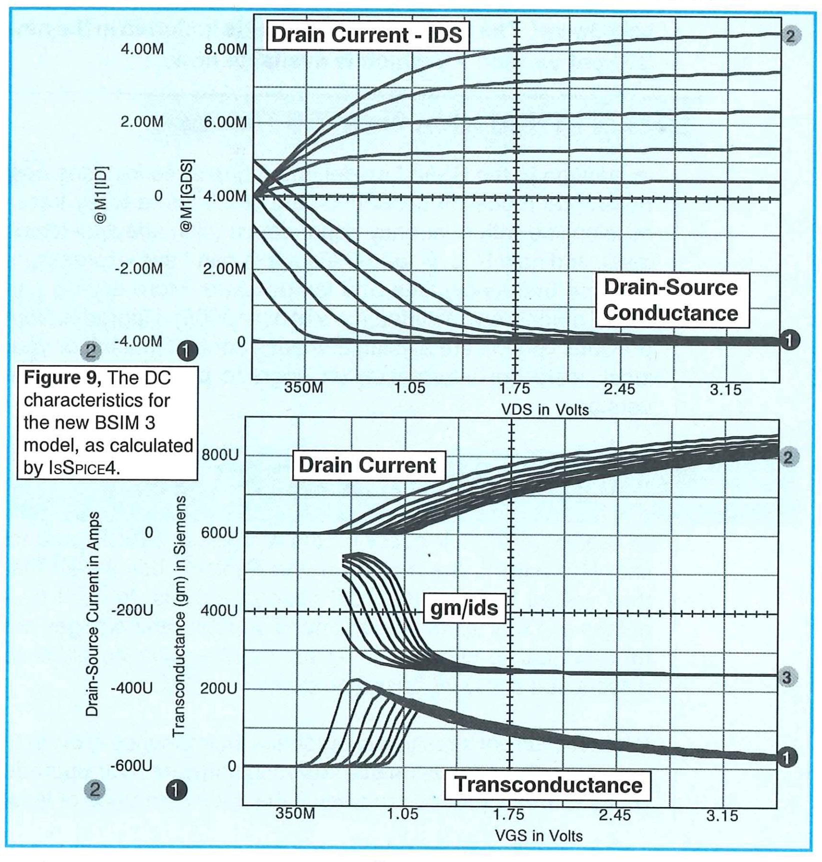


Figure 9, The DC characteristics for the new BSIM 3 model, as calculated by IsSpice4.

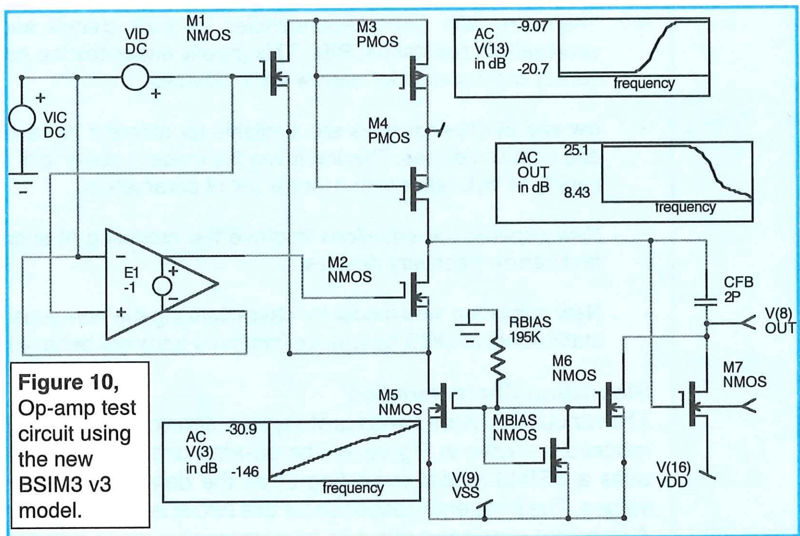


Figure 10, Op-amp test circuit using the new BSIM3 v3 model.

To obtain test circuits and more documentation about BSIM3V3 and join the BSIM3 users group mailing list, you can contact the BSIM3 homepage at: http://rely.EECS.Berkeley.EDU:8080/bsim3www/. **The new BSIM 3v3 model is included in the new IsSpice4 version 7.6 which is available now.**

BSIM3 Included in New 7.6 Release

In addition to the BSIM3 model, the 7.6 release includes new behavioral modeling constructs and functions, a lossy transmission line with frequency dependence (skin effect/dielectric loss), and now R, L, C, and B elements can have expressions that use frequency, time and temperature. More analog and digital models are included (now at over 8000). Upgrades from previous version are available. Please contact Intusoft or your local dealer for information on upgrade pricing for different versions.

Pspice Abandons DOS Users

In keeping with their policy on the Macintosh SPICE product line, Microsim™, the makers of the Pspice®, has stated that they will no longer offer DOS based products. Intusoft welcomes all DOS users who still need support, and pledges not to force you to upgrade to Windows. We have no plans to discontinue our DOS based products.

Intusoft does not force you to purchase maintenance in order to get full trade-up value for your existing software. Our upgrade policy is that upgrades are always the price difference or less!

Modeling Mechatronics

In previous *Intusoft Newsletters* [1], we have frequently addressed how to extend analog simulation into other domains. In the next few issues, we will explore how to model and simulate systems consisting of electrical, mechanical, and hydraulic elements. You might currently be using a mathematical tool like Mathcad or Matlab, or another non-SPICE simulator like Saber to analyze "mechatronic" problems, but as we'll see, IsSpice4 solves these problems just as efficiently with the added advantage that analog and digital electronics can be mixed in. Much of this work is adapted from J. Scholliers [2].

SPICE solves sets of nonlinear differential equations, and any problem that can be put into that form can be solved by SPICE. The added features of parametric and behavioral modeling and graphical input make IsSpice4 the MOST powerful and affordable multitechnical tool on the market today. It should be noted that IsSpice4 has a full AHDL capability, but in the case of mechatronics, this added power is not required.

Table 1 shows the mapping of "through" and "across" variables as they relate to their electrical counterparts. The energy conservation principle behind the relationships allows you to convert different systems to a single (electrical) domain. By translating the variables in the different domains to the electrical domain (voltage/current) any one or combination of the systems can be simulated with IsSpice. Table 2 shows what quantities the electrical elements represent. From the user's point of view, once the subcircuit and symbol have been constructed, the electrical analog remains hidden, and the simulation proceeds from a graphical perspective.

As an example of a mechatronic element, let's look at a model for a synchro. Synchros operate as rotating transformers, and

Table 1, Power variables in different energy domains.

	through	across	Power (P)
Electrical	current [A] i	e.m.f. [V] v	$i*v$
Magnetic	flux rate [Wb/s ≡V] $d\phi/dt$	m.m.f. [A] $\mathfrak{F}$	$\mathfrak{F}*d\phi/dt$
Mechanical Translational	force [N] F	velocity [m/s] $v=dx/dt$	$F*v$
Mechanical Rotation	torque [Nm] T	rot. velocity [rad/s] $\omega=d\theta/dt$	$\tau*\omega$
Fluidic	volume flow [m^3/s] Q	pressure [Pa] p	$Q*p$

Table 2, Electrical primitives in different energy domains. The units are S.I. (i.e. kg for mass, meter for length)

Electrical	Element Equation	Mechanical Translational	Mechanical Rotational	Fluidic
voltage source (V)	$v=f(t)$	velocity source (v)	velocity source (ω)	pressure source (p)
current source (I)	$i=f(t)$	force source (F)	torque source (T)	flow source (Q)
resistor (R)	$v=Ri$	damping (b)	rot. damping (b)	fluidic resistance
capacitor (C)	$i = C\, dv/dt$	mass (m)	inertia (J)	tank
inductor (L)	$v = L\, di/dt$	spring (k)	rot. spring (k)	fluidic inertia

are used in sensitive control applications where a mechanical output is required. The synchro is modeled as three transformers whose gains are dependent upon position. The gain of each transformer is derived from the position of the stator with respect to the rotor, which is equal to K*sin(α), K*sin(α+120), K*sin(α-120) for the synchro (3 windings at 120 degrees).

The rotor is modeled with a general purpose load model consisting of inertia, damping and Coulomb friction. This model

```
* SYNCHRO - Connections: s1,s2, s3: 3 nodes in star on the stator, r1, r2: 2 nodes for the rotor,
* w, wref: mechanical nodes. Parameters: K - gain, x0 - initial angle in radians, Jm - load inertia, b - load drag,
* w - load friction, v0 - initial mass velocity in m/s, d120 - 120 degrees in radians, K - gain

.Subckt Synchro s1 s2 s3 r1 r2 w wref {d120=2.094395}
xload w wref load {Jm=1f b=1f w=1f v0=0}
xint w wref theta posrel {x0=0}
beqsin1 qsin1 0 v=sin(v(theta))
beqsin2 qsin2 0 v=sin(v(theta) + {d120})
beqsin3 qsin3 0 v=sin(v(theta) - {d120})
x1 r1 r2 s1 n qsin1 0 transfov {k=1}
x2 r1 r2 s2 n qsin2 0 transfov {k=1}
x3 r1 r2 s3 n qsin3 0 transfov {k=1}
rn n 0 1T
.Ends
* LOAD (i.e. mass with connection to ground)
* In 101, Out 102, Gravity in m/s^2
.Subckt Load 1 2 {Jm=1 b=1n w=0 v0=0 g=0}
xm   1   massg {Jm=1f v0=0}
xd   1 2 damper {b=1f}
xw   1 2 frict1 {w=1f}
.Ends
* MASSG: mass with gravity
* Connections: 1 - X-component of velocity/force
.Subckt Massg 1 {Jm=1u  g=-9.81 v0=0}
v  1  101
r1 1 0  1T
r 101 0 1T
c 101 0 {Jm} ic={v0}
ig 0 1 {Jm*g}
.Ends
* DAMPER : Viscous damping, In 101, Ref 102
.Subckt DAMPER 101 102 {B=1}
r 101 102 {1/B}
r1 101 0 1T
r2 102 0 1T
.Ends
* FRICT: Constant friction force
* 1 2 : nodes between which force has to be added
.Subckt Frict1 1 2 {W=0}
rin     1 2 1T
befric  111 0  V= V(1,2)*{W}^0 < -.1m ? -1 :
+ V(1,2)*{W}^0 > .1m ? 1 : V(1,2)*{W}^0 /.1M
bgw  1 2 I = {W} * V(111)
.ENDS
* TRANSFOV: Ideal Transformator with variable ratio
* In-pri 101 102, Outsec 201 202, Control 301 302
.Subckt Transfov 101 102 201 202 301 302
+ {K=1 effi=1.00 effv=1.00}
rin 101 111 1n
be 205 202 v={K*effv} * v(301,302) * v(101,102)
bg 111 102 i={K} * v(301,302) * i(vm) / {effi}
vm 205 206
rp 101 102 1T
rs 206 201 1n
r3 301 302 1T
rout 201 202 1T
.Ends
* POSREL: Relative Position, Input 101(vel. of point)
* 102(ref. vel.), Output 201 (relative position of point)
* Parameters: x0 - initial position, K - gain
.Subckt Posrel 101 102 201 {x0=0 K=1}
Rin 101 102 1T
Bw1 103 0  v={K} * v(101,102)
Rtime 103 104 1MEG
Ctime 104 105 1U ic={x0}
R2 104 0 1G
Rf 104 105 1G
E2 0 105 104 0 1G
Rout 106 201 1N
Eout 106 0 0 105 1
R3 201 0 1T
.Ends
```

Figure 11, IsSpice4 netlist for a synchro model.

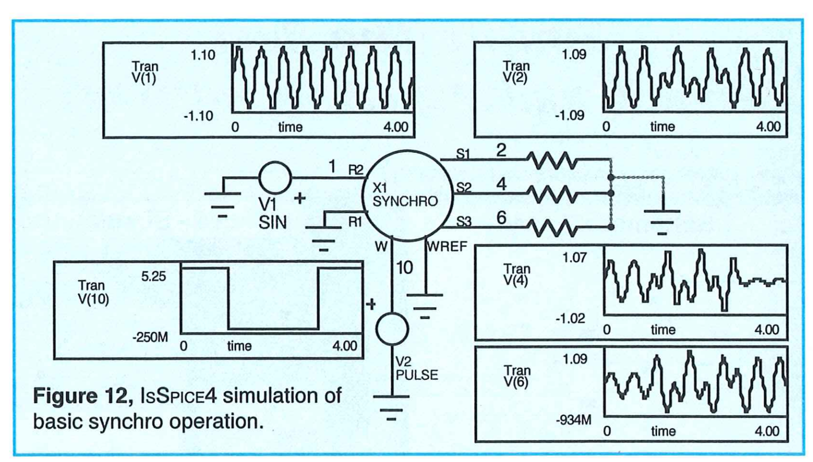


Figure 12, IsSpice4 simulation of basic synchro operation.

can be used in other mechanical components (e.g. the rotor in a motor, the piston in a hydraulic cylinder). The synchro has 5 electrical nodes (3 for stator windings, 2 for rotor windings), and 2 mechanical nodes. The parameters for the model are the gain (K) of the transformer, the initial angle (x0) of the rotor, and the rotor parameters (inertia-Jm, damping-b, friction-w and initial velocity-v0).

The rotor winding is excited by an AC reference voltage at frequencies ranging up to a few kHz. The magnitude of the voltage induced in any stator winding is proportional to the sine and cosine of the angle between the rotor-coil axis and the stator-coil axis. The voltage induced across a pair of stator terminals is the vector sum of the voltages across the two connected coils. If V sin(ωt) is the reference voltage in the rotor of a synchro, the stator voltages are: V1 = Vsin(ωt)Sin(α), V2 = Vsin(ωt)Sin(α+120) and V3 = Vsin(ωt)Sin(α+240), where α is the shaft angle.

When an AC reference signal drives the rotor winding, the amplitude of the voltage induced in the stator windings varies according to rotor angular position. Figure 12 shows the result of the step change in the mechanical input along with the resulting synchro output. A resolver model is also included on the newsletter subscriber's floppy disk.

In future newsletters, models for a variety of building blocks will be developed including DC and AC motors, hydraulics, control elements, and sensors. This research will culminate with a future release of a complete mechatronic library.

[1] "Modeling Motors", Intusoft Newsletter #19/20, October 1989
[2] Johan Scholliers, "Design and Simulation of Mechatronical Systems", VTT Automation January 1995

The Biggest & Best SPICE Libraries

What vendor has the most model types? What vendor has published 3 books on SPICE and modeling? What vendor has the only Windows based AHDL language? What vendor provides their own SPICE training? What vendor has the most behavioral modeling features? What vendor has a FREE modeling service for its customers?

No Contest - The only answer is Intusoft!

No SPICE Vendor can match the variety and quality of Intusoft.

- **Total Library Size Over 8000 parts, hundreds of model types**
- **User Defined C Code Models for analog, digital, array, and sampled-data elements**
- **AHDL Models using a Windows C based language**
- **Power Models (400+ models from Unitrode, Cherry, Linear Technology, Siliconix)**
- **RF Device Library (300+ models from H-P, NEC, Philips, Motorola)**
- **Vendor supplied IC/Op-Amp Models (1400+ models from Analog Devices, TI, APEX, etc.)**
- **More models created every day**

Purchasing and Technical Notes

System Packages: All ICAP packages are complete systems including SpiceNet, Model Libraries, one of the IsSpice versions, and IntuScope. ICAP/4Windows & ICAP/4Macintosh contain IsSpice4. ICAP/4Lite contains a reduced feature version of IsSpice4. ICAP/4Lite Xtra includes ICAP/4Lite plus the IntuScope and Model library Lite upgrades. ICAP/4DOS, ICAP/4PM (Power Mac) & ICAP/4M contain IsSpice3. ICAP/2 contains IsSpice 1.41. ICAP/4Windows and ICAP/4Lite runs on either Windows 3.1x, Windows95, or WindowsNT.

Deluxe option includes the following items: SpiceMod, RF Device Models OR Power Supply Designer's Library, all vendor supplied IC libraries, the "Spice Applications Handbook" and "A Spice Cookbook". The Deluxe option can be added to any ICAP package. SpiceMod is not included in the Macintosh Deluxe option. Each item can also be purchased separately.

CMSDK: The CMSDK is a toolkit for developing AHDL (C code) models. It requires WindowsNT or Windows95 and Microsoft Visual C++ 1.1 or 2.x. The CMSDK Bundle includes the toolkit and ICAP/4Windows.

Intusoft Technical Publications: The "Spice Applications Handbook" contains past Intusoft newsletters (from 6/86 to 1/94, 34 in all). The newsletters include Spice modeling information, simulation tips and techniques, and much more. "A Spice Cookbook" contains over 100 practical circuit examples encompassing a wide array of topics (RF, Power, Filters, Digital, Microwave). Each example includes a technical explanation, Spice simulation tips, netlist, schematic, and output graphs. The Spice models, schematics, and netlists from the "Spice Applications Handbook" or "A Spice Cookbook" are included on floppy disk when either is purchased. "Simulating With Spice" is included with all IsSpice programs except IsSpice3 and IsSpice4, which come with their own User's Guide.

Spice Model Libraries: All Spice model libraries come on floppy disk and are unencrypted. They may be purchased separately from the IsSpice simulator. Please specify Macintosh, DOS, or Windows when ordering.

Intusoft's World-Wide Support Staff

Listed below are dealers where Intusoft products, updates, information, and support may be obtained.

Thomatronik GmbH - Herbert M. Müller
Bruckenstr 1
D-83022 Rosenheim
Germany, Austria, Switzerland
Tele:8031-2175-0/FAX:8031-2175-30

Technology Sources Ltd. - Dr. Graham Plows
Grove House Lodge, Falmouth Ave
Newmarket Suffolk CB8 0LZ
UK, Denmark, Norway, Sweden, Italy, Greece
Tele: 01638-561460/FAX: 01638-561721

AMSC Co. Ltd. - Isao Tatsuno
6-7 Nakamachi 1-Chome
Musashino Tokyo 180
Japan
Tele: 422-54-6801/FAX: 422-56-1759
Macintosh Products

*OrCAD Japan - Stuart Harrington
2-19 Makado, Honmoku
Naka-ku, Yokohama 231
Japan
Tele: (045) 621-1911/FAX: (045) 621-2911
DOS, Windows, WindowsNT, and NEC

IDEMAX - Niels R Malotaux
Dorpsstraat 74
3732 HK De Bilt
Netherlands, Belgium
Tele: 030-2202-924/FAX: 030-2210-106

EXCEM - M Frederic Broyde
12, Chemin des Hauts de Clairefontaine
78580 Maule
France
Tele: 1-34-75-13-65/FAX: 1-34-75-13-66

Benchmark Systems Pte - K. Raghavan
151 Chin Swee Road, #06-13
Manhattan House, Singapore 0316
Malaysia, Singapore, Thailand
Tele: 65-737-1161/FAX: 65-737-6038

Cho Chieh Enterprise Ltd. - T. Lin
3F, NO.4 Lane 15, Chiu Chiung Str.
Lu Chou Hsiang, Taipei Hsien
Taiwan
Tele: 886-2-289-2605/FAX: 886-2-289-0151

Trident Systems - Praveen Kapoor
E-1/10, III Floor, East Patel Nagar
New Delhi 110 008
India
Tele: 11-578-7055/FAX: 11-576-7055

*EDAcom Tecnologia - Marcos Wesley
Rua Martim Fransico, 56
Sao Caetano do Sul Sp 09541-330
Brazil
Tele/FAX 55-11-441-4355

Eugene Data Inc. - C.C. Kim
Seo Bong B/D, 10-26, Non Hyeon-Dong
Kang Nam-Ku,
Seoul South Korea
Tele: 822-517-3707/FAX: 822-517-3706

Dong Am Corp. - Juesob Ju
Room 403 Chungjin Bldg.
54-2 Samsung-Dong, Kangnam-Gu
Seoul South Korea
Tele: 82-2-511-5300/FAX: 82-2-511-9682

Cadel Sp. z.o.o - Bogdan Olejarczyk
ul Tamka 38
00-355 Warsaw
Poland
Tele: (22) 27-50-61/FAX: (22) 635-52-62

Humansoft - Tamás Holman
H-1149 Budapest
Angol u. 24/b
Hungary
Tele: 36-1 1632879/FAX: 36-1 2513673

Cadforce Systems Inc. - (Main Office)
Canada - (Montreal, Ottawa, Toronto)
55 Place Frontenac
Pte - Claire PQ
H9R4Z7 Canada
Canada - Regis Harrison
Toll Free (800) 249 7332
Tele: 514-426-2777/FAX: 514-426-0266
Canada - Ontario, Monique Massery
Tele: 905-469-0442/FAX: 905-469-0443

DS-Design Systems OY-Hannu Tikkanen
Sepankatu 4
SF-40100 Jyvaskyla
Finland
Tele: 41-652588/FAX: 41-610725

ME Technologies - Peter Stein
P.O. Box 50, Wang Wauk Rd.
Dyers Crossing, NSW 2429
Australia
Tele: (0)65-50-2254/FAX: (0)65-50-2341

*New Name, Address, or Phone Number